ME 566
Computational Fluid Dynamics for Fluids Engineering Design
CFX - 5 STUDENT USER MANUAL

Gordon D. Stubley
Department of Mechanical Engineering, University of Waterloo

©G.D. Stubley 2004,2005

2005

Contents

1 Student User Manual 1
- 1.1 Getting Started 2
  - 1.1.1 Windows XP/NEXUS 2
  - 1.1.2 Introduction to Workbench 2
- 1.2 The Problem 6
- 1.3 The CFD Model Specification 6
- 1.4 Geometry and Mesh Specification 8
  - 1.4.1 Basic Concepts and Definitions 8
  - 1.4.2 Geometry Creation 9
  - 1.4.3 CFX-Mesh: Mesh Generation 10
- 1.5 CFX-Pre: Physical Modelling 14
  - 1.5.1 Domain 14
  - 1.5.2 Initialization 18
  - 1.5.3 Output Control 18
  - 1.5.4 Simulation Type 18
  - 1.5.5 Solver Control 18
- 1.6 CFX Solver Manager: Solver Operation 19
  - 1.6.1 Monitoring the Solver Run 19
- 1.7 CFX-Post: Visualization and Analysis of Results 21
  - 1.7.1 Objects 21
  - 1.7.2 Tools 23
  - 1.7.3 Controls 23
- 1.8 Commands for Duct Bend Example 24
  - 1.8.1 Geometry Model 24
  - 1.8.2 Mesh Generation 28
  - 1.8.3 Pre-processing 29
  - 1.8.4 Solver Manager 34
  - 1.8.5 Post-processing 35
  - 1.8.6 Clean Up 37

1
Student User Manual

In these notes the basic steps in a CFD solution will be illustrated using the professional software ANSYS Workbench Version 9.0 Service Pack 1 which includes the components DesignModeler, CFX-Mesh, and CFX-5 (Version 5.7.1 Service Pack 1) (all trademarks of ANSYS). These notes include an introductory tutorial and a mini user's guide. They are not meant to replace a detailed user's guide. For full information on these components refer to the on-line help documentation provided with the software[1].

These notes include sections on:

Getting Started: Instructions for a short computer session in which the software graphical user interfaces, GUIs, are introduced;

The Problem: A description of the example problem;

The CFD Specification: A complete description of the CFD model implemented in the software;

Software Components: A description of the concepts and operation involved in the five software components: DesignModeler, CFX-Mesh, CFX-Pre, CFX-Solver, and CFX-Post; and

Commands for the Example Problem: A complete step-by-step list of instructions for solving the model problem.

The following font/format conventions are used to indicate the various commands that should be invoked:

Menu/Sub-Menu/Sub-Sub-Menu Item chosen from the menu hierarchy at the top of a main panel or window,

Button/Tab Command Option activated by clicking on a button or tab,

Link description Click on the description to move by a link to the next step/page;

Name *value* Enter the *value* in the named box,

Name ▼ **selection** Choose the selection(s) from the named list,

Name Panel or window name,

☐ **Name** On/off switch box, and

◯ **Name** On/off switch circle (radio button).

[1]Many of the features available in these software components will not be explored in introductory CFD courses.

1.1 Getting Started

This working session has two purposes:

1. to ensure that your Windows XP/NEXUS operating system is operational, and
2. to introduce the look and feel of the software.

1.1.1 Windows XP/NEXUS

The CFD software is available on the workstations in the all Engineering Computing labs, Fulcrum (E2-1313), Wheel (E2-1308), Lever (E2-1302), Wedge (E2-1302B), Helix (RCH-108), and GAFF (CPH-2367), and the Mechanical Engineering 4th year computing room, E2-2354. The workstations use the Windows XP operating system on Waterloo NEXUS. You should be familiar with techniques to create new folders (or directories), to delete files, to move through the folder (directory) system with Windows Explorer, to open programs through the Start menu on the Desktop toolbar, to move, resize, and close windows, and to manage disk space usage with tools like WinZip.

1.1.2 Introduction to Workbench

The ANSYS Workbench environment provides an interface to manage the files and databases associated with the individual software components. These files and databases are organized into a particular project. To get a feel for this environment and the GUIs associated with the software components, we will look at a pre-prepared project on flow through a pipe bend.

1. Create a working directory called CFDTest on your N drive.
2. Use a web browser to visit the UW-ACE ME 566 course page (uwace.uwaterloo.ca). Under the Lessons tab and open the Student User Manual folder. Click on the link to PipeBend.zip and follow the instructions to download the archive containing the working files.
3. Use WinZip to extract the files in the PipeBend archive into your working directory.
4. Open ANSYS Workbench from **Start/Programs/Engineering/ANSYS 9.0/ANSYS Workbench**.
5. Open the project file. Check that **Open:** ▼ Workbench Project is selected before using the **Browse** button below the **Open: Workbench Projects** panel area to find and selecting the file PipeBend.wbdb from your working directory.
6. There are three main areas on the screen: command menus, buttons, and tabs at the top, a list of potential Project Tasks on the left, and a list of the files linked to the project.
7. On-line help for Workbench, DesignModeler, and CFX-Mesh is available in web-page format similar to other Windows programs. Choose **Help/ANSYS Workbench Help** to open the ANSYS Workbench Documentation. Search

for **keyword** Tutorials and select CFX-Mesh Help. Follow the Tutorials link to see the list of available tutorials.

Introduction to DesignModeler/CFX-Mesh GUI

1. In the Workbench file area click on the filename PipeBend just to the right of the DesignModeler button (**DM**). Notice that the Project Tasks area at the left adjusts to reflect your choice. Under DesignModeler Tasks choose Open to open the geometry file.
2. A new page for DesignModeler will open. Go back to the Project page by clicking on the **PipeBend [Project]** tab at the top left of the screen. Click on the **PipeBend [DesignModeler]** tab to return to the DesignModeler page.
3. There are four major areas on the page: command menus and buttons at the top, a Tree View and Sketch Toolbox on the left, a Details View at the bottom left, and a Model View window. Place the mouse cursor over one of the command buttons in the top row. A brief description of the button's action should appear (you may need to click in the window once to make it active). Visit each button with the mouse cursor to see its action.
4. One method of controlling the view is with the coordinate system triad in the lower right corner of Model View. Click on the Z axis of the triad to see a back view of the pipe bend. Click on the cyan sphere to select the isometric view.
5. Another method of controlling the view is with the mouse left button in conjunction with a mouse action selection. From the upper row of buttons select the **Pan** action. Holding the left mouse button down drag the mouse over the Model View to translate the view. Select the **Zoom** action and repeat the mouse action to change the size of the view.
6. Select the **Rotate** action. The rotate action is context sensitive in that it depends upon the position of the mouse cursor. With the mouse cursor close to the pipe bend, press the left mouse button to get *free 3D rotation*. The point of rotation can be changed by clicking the left button while the cursor is on the pipe bend surface (this may take some experimentation). A small red sphere indicates the centre of rotation. With the mouse cursor in a corner of the Model View, press and hold the left mouse button get a *roll* action in which there is 2D rotation about an axis perpendicular to the Model View window. Move the cursor to either the left or right of the Model View and hold the left button to get a *yaw* action in which there is 2D rotation about the horizontal axis. Move the cursor to either the top or bottom of the Model View and hold the left button to get a *pitch* action in which there is 2D rotation about the vertical axis.
7. Rotate, zoom, and pan actions can be achieved directly by pressing the middle mouse key alone, with the `Shift` key, and with the `Ctrl` key, respectively.

8. The Tree View on the left shows the geometric entities that were used to generate the cylinder. Expand the 1 Part, 1 Body entity and click on Solid to see some properties of the cylinder in the Details View. The pipe bend was generated from two entities:
   (a) **Sketch1:** which can be found in the Plane4 entity. Click on Sketch1 to highlight the circle that the pipe bend is based upon with yellow.
   (b) **Sketch2:**. which can be found in the YZPlane entity. Click on the Sketch2 to highlight the path that is swept out by Sketch1 to generate the pipe bend.
9. In the help page search for **keywords** rotation modes and select CFX-Mesh Help to find more information on changing the view.
10. Click the **X** button on the **PipeBend [DesignModeler]** tab to close the DesignModeler page. You can click No to quit without saving changes.

Introduction to the CFX-Post GUI

The tasks associated with CFD simulation in Workbench are referred to as Advanced CFD Tasks. For historical reasons, there are differences in the GUI and Help files for the three CFX-5 components. We will use CFX-Post to look at the completed simulation of flow through a pipe bend to get a feel for these interfaces.

1. In the Workbench file area click on the filename PipeBend_001. Under Advanced CFD Tasks choose Open in CFX-Post to open the results file (PipeBend_001.res). All of the pertinent CFD model data (mesh, flow attributes, and boundary condition information) for this problem is stored in this file.
2. A new page for CFX-Post will open.
3. There are three major areas on the screen: Command menus and buttons at the top, selector and edit panels on the left, and **Viewer** window. Wireframe models of the pipe inlet and outlet should be in the **Viewer**. To see the pipe bend click the ☐ **Domain 1 Default** object on in the Objects panel.
4. The mouse button action for controlling the view is similar to that in DesignModeler. There is a significant difference in that the rotation action is not context sensitive. 2D rotation about an axis perpindicular to the screen is achieved by pressing the `Ctrl` key with the left button while in rotation mode.
5. The coordinate system triad is shown in the lower right corner. Unlike in DesignModeler, the triad cannot be used to change the view. The view can be changed with the view buttons at the top of the Viewer window. Use the **-X** to select an orthographic view towards -x along the x axis.
6. Now look at some results. Each visualization is created by defining a new visualization object and then editing the properties of the object. To make a vector plot along the centre plane of the pipe bend:

(a) Choose **Create/Object/Plane** from the top menu row (or click on the [**Create Plane**] button. In the **New Plane** panel accept **Name** [Plane 1], and click [**Ok**] to open a **Plane** edit panel in the lower left. In the edit panel set:

- **Domains** ▼ [All Domains] (the default),
- **Method** ▼ [YZ Plane], **X** [0], ☐ **Visibility** on

and click [**Apply**]. To see this plane, click the ☐ **Domain 1 Default** object off in the Objects panel.

(b) Choose **Create/Object/Vector** from the top menu row (or use the corresponding button on the second row). In the **New Vector** panel accept **Name** [Vector 1], and click [**OK**] to open a **Vector** edit panel. In the edit panel set:

- **Locations** ▼ [Plane 1],
- **Variable** ▼ [Velocity],
- ◯ **Hybrid** on,
- **Projection** ▼ [None],
- **Reduction** [10],
- ☐ **Visibility** on

and click [**Apply**]. The vector plot should appear in the **Viewer** window and the vector object should be listed in the **Object** database panel. Under the **View Control** branch in the **Object** tree, turn ☐ **DEFAULT LEGEND** off and back on to remove and then replace the scale legend.

7. To visualize the pressure field, create a contour plot. Remove the vector plot, click ☐ **Vector 1** and ☐ **Plane 1** off in the **Object** tree. Choose **Create/Object/Contour** from the top menu row. In the **New Contour** panel accept **Name** [Contour 1], and click [**OK**]. In the **Contour - Contour 1** edit panel set:

- **Locations** ▼ [Plane 1],
- **Variable** ▼ [Pressure],
- ◯ **Hybrid** on,
- **Colour Scale** ▼ [Linear],
- **Colour Map** ▼ [Rainbow],
- **# of Contours** [10],
- ☐ **Visibility** on

and click [**Apply**]. The fringe plot should appear in the **Viewer** window. Go to the [**Render**] tab in the **Contour** edit panel , turn ☐ **Draw Faces** off, and click [**Apply**] to see a line contour plot of the pressure field. Does this pressure field make sense to you?

8. On-line help is available in pdf file format, **Help**. Context-sensitive help is also available. Position the mouse pointer in the **Object Selector** panel and

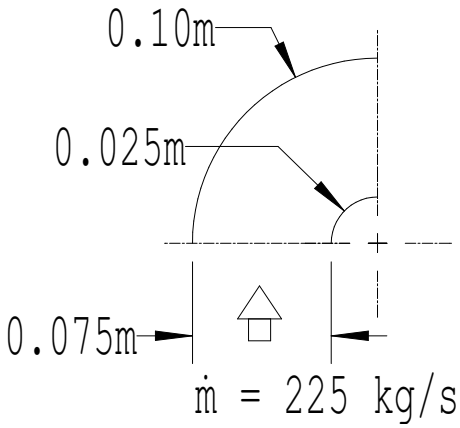

Fig. 1.1 Geometry of short radius duct bend

press **<F1>** to bring up the help page for that panel.

9. This should give a sense of the operation of the CFX GUI. Feel free to experiment with other object types and scalar fields. When you have finished, return to the Workbench Project page. Exit by **File/Close Project** and choose No: do not save any items.
10. Clean up by deleting the CFXTest directory.

1.2 The Problem

Consider the analysis problem of estimating the pressure drop of flow through the short radius duct bend shown in Figure (1.1). The duct bend has a width of $1[m]$ and is made of galvanized steel with an average surface roughness height of $0.10[mm]$. Water flows through the bend with a mass flow rate of $\dot{m} = 225[kgs^{-1}]$.

1.3 The CFD Model Specification

The first phase in the CFD solution is a planning stage in which the complete CFD model of the duct bend is specified. This specification includes:

Physical Model Specification The steel walls of the bend and other duct pieces are assumed to be rigid and joints in the duct work are assumed to be smooth. The galvanized steel is assumed to have a uniform surface roughness height. The width of the bend is sufficient that the flow can be considered to be two-dimensional.

Domain Geometry Specification To ensure that reasonable flow patterns are simulated in the bend it is necessary to add short entrance and exit lengths of duct to simulate the actual flow through the bend when it is situated in a duct. The domain geometry is shown in Figure (1.2).
The CFD simulation code is fully three-dimensional, so even though we are primarily interested in flow in the plane shown in Figure (1.2), the geometry

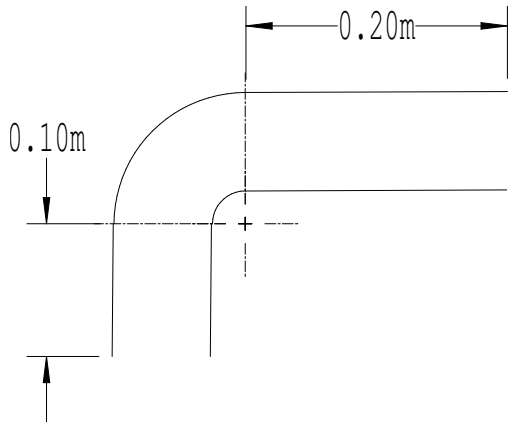

Fig. 1.2 Geometry of the duct bend model

model must have a width into the page. A thin slice is used. The width of the slice is set to $0.02[m]$.

Specification of Simulation Type For this application a *steady* flow is appropriate.

Specification of Fluid Properties For this application the fluid is water which can be treated as a simple liquid with nominal constant properties ($\rho = 1000[kgm^{-3}], \mu = 0.001[Pa * s]$).

Specification of Flow Models For this analysis it is reasonable to assume the following flow features:

- incompressible flow,
- fully turbulent flow (the Reynolds number is approximately 225,000),
- the turbulent momentum stresses can be modelled with the standard $k - \varepsilon$ model:

$$\tau_{ij} = \mu_t \left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j} \right) \tag{1}$$

  where the turbulent viscosity, μ_t, is proportional to the fluid density, the velocity scale of the turbulent eddies and the length scale of the eddies. The scales of the turbulent eddying motion are estimated from two field variables which are calculated as part of the model: k, the turbulent kinetic energy, and ε, the rate at which k is dissipated by molecular viscous action.

Specification of the Boundary Conditions The boundary conditions that model the interaction of the surroundings with the solution domain are:

- uniform velocity of $3[ms^{-1}]$ and uniform turbulence properties of turbulence intensity of 5% and turbulence eddy length scales of $0.0075[m]$ (i.e. 10% of the duct height) across the inlet surface,
- uniform static pressure across the outlet surface,

- no-slip conditions along the duct walls and the standard wall-function treatment to resolve log-law behaviour in the near wall region where the flow is not fully turbulent, and
- symmetry conditions on the front and back surfaces (to ensure that the simulated flow is two-dimensional).

The above provides a mathematically complete description of the CFD model. In the next four sections, information will be provided on the use of the software that can implement the CFD steps of geometry modelling, mesh generation, physical modelling, solver operation, and visualization. The actual software commands to use for this example problem are given in Section 1.8.

1.4 Geometry and Mesh Specification

In the first steps of the CFD computer modelling, the solution domain is created in a digital form and then subdivided into a large number of small finite elements or volumes. Common finite element types (shapes) include: *tetrahedral, prismatic*, and *hexahedral*. These notes present a basic methodology for developing simple geometries for tetrahedral meshes. To develop expertise for more complex geometries work through the sequence of tutorials in given in CFX-Mesh 2.1 Help.

1.4.1 Basic Concepts and Definitions

Vertex: Occupies a point in space. Often other geometric entities like edges *connect* at vertices.

Edge: A curve in space. An *open edge* has beginning and end vertices at distinct points in space. A straight line segment is an open edge. A *closed edge* has beginning and end vertices at the same point is space. A circle is a closed edge.

Face: An enclosed surface. The surface area inside a circle is a *planar face* and the outer shell of a sphere is a *non-planar face*. An *open face* has all of its edges at different locations in space. A rectangle makes an open face. A *closed face* has two edges at the same location in space. The cylindrical surface of a pipe is a closed face.

Solid: The basic unit of three dimensional geometry modelling:

- is a space completely enclosed in three dimensions by a set of faces (*volume*);
- the surface faces of the solid are the the external surface of the flow domain; and
- holes in the solid represent physical solid bodies in the flow domain such as airfoils, etc.

Part: One or more solids that form a flow domain.

Multiple Solids: May be used in each part:

- the solid volumes cannot overlap;
- the solids must join at common surfaces or faces; and

- the faces where two solids join can be *thin surfaces*

Thin Surface: A thin solid body in a flow like a guide vane or baffle can be modelled as an infinitely thin surface with no-slip walls on both sides.

Units: To keep things simple and to minimize errors, use metric units throughout.

Advanced Concepts: See the Geometry section of the CFX-Mesh Help for further information on geometry modelling requirements. To develop improved skill follow the tutorials given in CFX-Mesh Help/Tutorials.

1.4.2 Geometry Creation

The basic procedure for creating a three dimensional solid geometry is to make a 2D sketch of an enclosed area (possibly with holes) on a flat plane. The resulting 2D sketch is a profile which is swept through space to create a 3D solid feature. This process can be repeated to either remove portions of the 3D solid or to add portions to the solid.

Each sketch is made on a *Plane*:

- There are three default planes, XYPlane, XZPlane, and YZPlane, which coincide with the three planes of the Cartesian coordinate system;
- Each plane has a local X-Y coordinate system and normal vector (the plane's local Z axis);
- New planes can be defined based on: existing planes, faces, point and edge, point and normal direction, three points: origin, local X axis, and another point in plane, and coordinates of the origin and normal; and
- Plane transforms such as translations and rotations can be used to modify the base definition of the plane.

The creation of a sketch is similar to the creation of a drawing with modern computer drawing software:

- A sketch is a set of edges on a plane. A plane can contain more than one sketch;
- The sketching toolbox contains tools for drawing a variety of common two dimensional shapes;
- *Dimensions* are used to set the lengths and angles of edges;
- *Constraints* are used to control how points and shapes are related in a sketch. Common constraints include:

  Coincident (C): The selected point (or end of edge) is coincident with another shape. For example, the end point of a new line segment can be constrained to lie on the line extending from an existing line segment. Note that the two line segments need not touch;

  Coincident Point (P): The selected points are coincident in space;

  Vertical (V): The line is parallel to the local plane's Y axis;

  Horizontal (H): The line is parallel to the local plane's X axis;

  Tangent (T): The line or arc is locally tangent to the existing line or arc;

  Perpendicular ($\perp$): The line is perpendicular to the existing line; and

Parallel (∥): The line is parallel to the existing line.
As a sketch is drawn the symbols for each relevant constraint will appear. If the mouse button is clicked while a constraint symbol is on the sketch then the constraint will be applied. Note that near the X and Y axes it is often difficult to distinguish between coincident and coincident point constraints; and

- *Auto-Constraints* are used to automatically connect points and edges. For example, if one edge of a square is increased in length the opposite edge length is also increased so that the shape remains rectangular.

Features are created from sketches by one of the following operations:

Extrude: Sweep the sketch in a particular direction (i.e. to make a bar);
Revolve: Sweep the sketch through a revolution about a particular axis of rotation (i.e. to make a wedge shape);
Sweep: Sweep the sketch along a sketched path (i.e. to make a curved bar); and
Skin/Loft: Join up a series of sketches or profiles to form the 3D feature (like putting a skin over the frame of a wing).

Features are integrated into the existing active solid with one of the following Boolean operations:

Add Material: Merge the new feature with the active solid;
Cut Material: Remove the material of the new feature from the active solid;
Slice Material: Remove a section from an active solid; and
Imprint Face: Break a face into two parts. For example, this will open a hole on a cylindrical pipe wall.

Sometimes it is necessary to use multiple solids in a single part. These solids must share at least one common face. This common face might be used to model a thin surface in the flow solver. In this case:

1. Select active solid with the body selection filter turned on;
2. Freeze the solid body to stop the Boolean merge or remove operations (**Tools/Freeze**). This will form a new solid body as a component of a new part; and
3. Select all solids and choose **Tools/Form New Part**.

When the solid model is completed an .agdb file is created and saved in order to store the geometry database.

1.4.3 CFX-Mesh: Mesh Generation

The mesh generation phase can be broken down into the following steps:

1. Read in or update the .agdb file with the solid body geometry database;
2. Set the properties of the mesh;
3. Cover the surfaces of the solid body with a surface mesh of triangular or quadrilateral elements; and
4. Fill the interior of the solid body with a volume mesh of tetrahedral, hexahedral, or prism elements that are based on the surface meshes and that

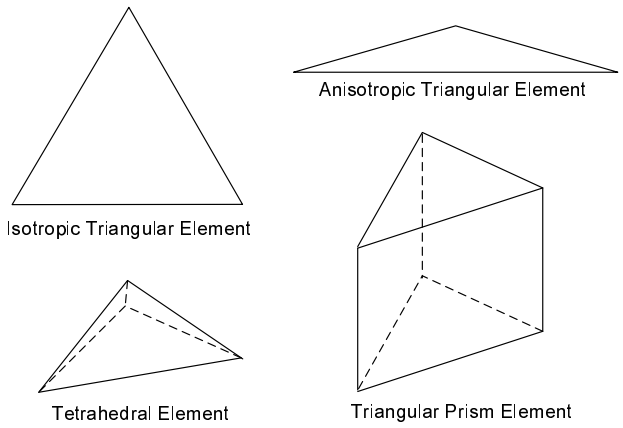

Fig. 1.3 Shape of common two and three dimensional elements.

grow inwards from each surface mesh. A .gtm file containing all of the mesh information and region information is written at the end of this step.

The following comments and guidelines are for generating meshes for two-dimensional flow simulation in relatively simple rectangular geometries.

Regions

The geometry database contains a list of primitive faces and edges that are formed in the generation processes. It is often cumbersome to work directly with these primitive entities. Therefore, there is a facility for creating and naming composite 2D surface regions. When the mesher is initialized all of the primitive 2D surfaces are assigned to the Default 2D Region. As surfaces are assigned to new user defined regions they are removed from the Default 2D Region. However, the Default 2D Region must contain at least one surface. These region names and the their corresponding surface meshes are passed on to CFX-Pre.

Mesh Features

The mesh is composed of two dimensional triangular elements on the surfaces and tetrahedral and prism elements in the body of the solid. Figure (1.3) shows the element shapes.

The properties of the mesh are controlled by the settings of the following features:

Default Body Spacing: Set the maximum length scale of the tetrahedral elements throughout the volume of the body. Some of the actual tetrahedral elements may be smaller due to the action of other mesh features or in order to fit the tetrahedral elements into the body shape.

Default Face Spacing: Set the length scale of the triangular elements on the surfaces.

- For simple meshes it is sufficient to set **Face Spacing Type ▼** Volume Spacing.

- For surfaces such as an airfoil in a large flow domain, it might be desirable to set the triangular mesh length scale smaller than the default body spacing. In this case a new Face Spacing can be defined and assigned to the airfoil surface. Besides setting the triangular element length scale, the following properties must be set for the new face spacing:
  Radius of Influence: The distance from the region that has a tetrahedral mesh length scale equal to that of the surface triangular elements;
  Expansion Factor: The rate at which the tetrahedral mesh length scale increases outside the radius of influence. This value controls how smoothly the mesh length scale increases from the face region to the default body spacing far from from the face.
- For complex surfaces the face spacing type should set so that the geometry of the surface is well represented by the mesh: relative error or angular resolution.

Controls: are used to locally decrease the mesh length scale in the region around a point, line, or triangular plane surface. The spacing in the vicinity of a control is set by three factors:
Length Scale: fixes the size of the tetrahedral mesh elements;
Radius of Influence: sets the distance from the control that has a mesh of the specified length scale; and
Expansion Factor: controls how smoothly the mesh length scale increases to the default body spacing far from the control.
For line and triangle controls, the spacing can be varied over the control (i.e. from one end point of the line to the other end point).

Extruded Periodic Pair: In cases where the flow is two dimensional, it is desirable to have a single mesh element in the cross-stream direction. In these cases the surface meshes on two surfaces will need to be identical (i.e. the face mesh on the first surface can be uniquely mapped onto the second surface). In other cases where the flow is three dimensional, it may still be desirable to have identical face meshes on two bounding surfaces. For example, this is useful in periodically repeating geometries.
Each Periodic Pair is defined by two surfaces and a two dimensional planar (**Periodic Type** ▼ Translational) or axisymmetric (**Periodic Type** ▼ Rotational) mapping.

Inflation: In boundary layer regions adjacent to solid walls it is often desirable to make a very small mesh size in the direction normal to the wall in order to resolve the large velocity shear strain rates. If tetrahedral meshes are used in this region there will either be a large number of very small elements with equal spacing in all directions (i.e. *isotropic* elements with vertex angles close to 60°) or very thin squashed elements. These choices are either inefficient or inaccurate. A better element shape in this region is a triangular prism, Figure (1.3), based on the surface triangular mesh. The basic shape of the

prism element is independent of the height of the prism (mesh length scale normal to the wall). The layer of prism elements is an inflated boundary with:

Maximum Thickness: that is often approximately the same as the default body mesh spacing; or

First Layer Thickness: that is often set by the properties of the local turbulent boundary layer. Other properties include:

Number of Inflated Layers: specifies the number of prism elements across the thickness of the inflated layer; and

Expansion Factor: specifies how the prism height increases with each inflated layer above the wall surface. This factor must be between 1.05 and 1.35.

Stretch: The default body mesh length scale is isotropic. The vertex angles in the isotropic tetrahedral elements are close to 60°. In geometries that are not roughly square in extent, it may be desirable make the mesh length scale longer or shorter in one particular direction. This is achieved by stretching the geometry in a given direction, meshing the modified geometry with an isotropic mesh, and then returning the geometry (along with the mesh) to its original size. This means that if the y direction is stretched by a factor of 0.25 without stretching in the other two directions then the mesh size in the y direction will be roughly 4 times that of the other directions. Take care to ensure that the resulting tetrahedral elements do not get too squashed. For this reason the stretch factors should be between 0.2 and 5 at the very most (more moderate stretch factors are desirable). **Note:** stretch parameters are ignored in extruded meshes.

Proximity: flags set the behaviour of the mesh spacing when edges and surfaces become close together. For simple rectangular geometries set **Edge Proximity** ▼ No and **Surface Proximity** ▼ No.

Options: are used for setting the output filename and for setting the algorithms used for generating the volume and surface meshes:

Volume Meshing ▼ Advancing Front is the primary algorithm for generating meshes in 3D geometries. The algorithm starts with a surface mesh and then builds a layer of tetrahedral elements over the surface based on the surface triangular elements. This creates a new surface. The process is repeated advancing the layers of tetrahedral elements into the interior of the volume.

Volume Meshing ▼ Extruded 2D Mesh is an algorithm for generating meshes in geometries that are effectively 2D. A surface mesh is extruded through space by either translation or rotation from one face to a matching face in the periodic pair. This option is useful for simulating two dimensional flows and flows in long constant area ducts. The number of elements (often 1) and mesh spacing distribution in the extruded direction can be specified.

Surface Meshing ▼ Delaunay is a fast algorithm for creating isotropic

surface meshes. Suited to complex surface geometries with small mesh spacing.

Surface Meshing ▼ Advancing Front starts at the edges of the surface and is a similar algorithm to the volume meshing algorithm described above. Since it creates regular meshes on simple rectangular type surfaces , it is the recommended algorithm.

1.5 CFX-Pre: Physical Modelling

CFX-Pre is a program that builds up a database for storing all of the information (geometry, mesh, physics, and numerical methods) that is required by the equation solver. The contents of the database is written to a `def` (definition) file at the end of the CFX-Pre session.

The database is organized as a hierarchy of objects. Each object in the hierarchy is composed of sub-objects and parameters. There are two main objects: **Flow** and **Library**. The **Flow** object holds all of the data on the flow model and the **Library** object holds the property data on a set of common fluids.

The major components of the **Flow** object are organized in the following hierarchy:

- **Flow**
  - **Domain**
    - ***Fluids List***
    - **Boundary**
    - **Domain Models**
      - **Domain Motion**
      - **Reference Pressure**
    - **Fluid Models**
      - **Heat Transfer Model**
      - **Turbulence Model**
      - **Turbulent Wall Functions**
  - **Initialization**
  - **Output Control**
  - **Simulation Type**
  - **Solver Control**
    - **Advection Scheme**
    - **Convergence Control**
    - **Convergence Criteria**

CFX-Pre has functions to create new objects in the hierarchy and to edit existing objects through edit panels. For most objects, the edit panel provides guidance on the possible parameter settings.

1.5.1 Domain

Fluids List

A fluid (or mixture of fluids in more complex multi-phase flows) has to be associated with each domain. The fluid for a particular domain can be selected from

the fluid library which has many common fluids. The **Materials** panel manages the fluid library. There are provisions for selecting pre-defined fluids, defining new fluids, and creating duplicates or copies of existing fluids.

The properties of fluids can be general functions of temperature and pressure for liquids or gases. The **C**FX **E**xpression **L**anguage, **CEL** is used to input formulae for specifying equations of state and other applications such as boundary variable profiles, initialization, and post-processing. **CEL** allows expressions with standard arithmetic operators, mathematical functions, standard CFX variables, and user-defined variables. All values must have consistent units and variables in **CEL** expressions must result in consistent units. Full details of **CEL**, including the names of the standard CFX variables, are included in the *CFX-5 Reference Material/Guide*.

For low speed flows of gases and liquids it is adequate to use constant property fluids. It is easiest to build up a new constant property fluid from a comparable fluid from the existing CFX fluid library as a template. For example, to define constant property air at 20 $^{\circ}C$, make a duplicate copy of air at 25 $^{\circ}C$ and then edit this copy.

Boundary

Throughout each domain, mass and momentum conservation balances are applied over each element. These are universal relationships which will not distinguish one flow field from another. To a large extent a particular flow field for a particular geometry is established by the boundary conditions on the surfaces of the domain.

A standard boundary condition object includes a name, a type, a set of surfaces, and a set of parameter values. In CFX-Post the boundary condition object name is used to refer to the set of surfaces on which the boundary condition is applied. For this reason the habit of naming each boundary condition by the name of its surfaces (as defined in CFX-Mesh) is often followed.

Boundary condition types include:

Inlet: an inlet region is a surface over which mass enters the flow domain. For each element face on an inlet region, one of the following must be specified:

- fluid speed and direction (either normal to the inflow face or in a particular direction in Cartesian coordinates),
- mass flow rate and flow direction, or
- the total pressure -

$$P_{total} \equiv P + \frac{1}{2}\rho V^2 = P_{total\,spec} \tag{2}$$

and flow direction

If the flow is turbulent then it is necessary to specify two properties of the turbulence. Most commonly, the intensity of the turbulence -

$$I \equiv \frac{\texttt{Average of speed fluctuations}}{\texttt{Mean speed}} \tag{3}$$

and one additional property of the turbulence: the length scale of the turbulence (a representative average size of the turbulent eddies), or eddy viscosity ratio (turbulent to molecular viscosity ratio, μ_t/μ) are specified. Typical turbulence length scales are 5% to 10% of the width of the domain through which the mass flow occurs.

Outlet: an outlet region is a surface over which mass leaves the flow domain. For each element face on an outlet region, one of the following must be specified:

- fluid velocity (speed and direction),
- mass flow rate, or
- static pressure

A specified static pressure value can be set to a specific face, applied as a constant over the outflow region, or treated as the average over the outflow region. No information is required to model the turbulence in the fluid flow at an outflow.

Opening: a region where fluid can enter or leave the flow domain. Pressure and flow direction must be specified for an opening region. If the opening region will have fluid entering/leaving close to normal to the faces (i.e. a window opening) then the specified pressure value is the total pressure on inflow faces and the static pressure on outflow faces (a mixed type of pressure). If the opening region will have fluid flow nearly tangent to the faces (i.e. the far field flow over an airfoil surface) then the specified pressure is a constant static pressure over the faces. For turbulent flows, the turbulence intensity must also be set.

Wall: a solid wall through which no mass can flow. The wall can be stationary, translating (sliding), or rotating. If the flow field is turbulent then the wall can be either smooth or rough. Depending upon which of these options are chosen, suitable values must be input (i.e. the size of the roughness elements, etc.).

Symmetry: a region with no mass flow through the faces and with negligible shear stresses (and negligible heat fluxes). This condition is often used to simulate a two-dimensional flow field with a three-dimensional flow solver and to minimize mesh size requirements by taking advantage of natural symmetry planes in the flow domain.

Since it is crucial that each surface element face have a boundary condition attached to it, CFX-Pre automatically provides a default boundary condition for each domain. Once all boundary surfaces have been attached to explicit boundary conditions, the default boundary condition object is deleted. This allows the user to identify surfaces which still require explict boundary conditions.

For the flow solver to successfully provide a simulated flow field, the specified boundary conditions should be realizable (i.e. they should correspond to conditions in a laboratory setup). In particular, ensure that the inlet and outlet boundary conditions are consistent and that they take advantage of the known information. Table (1.1) lists several common inlet/outlet condition combinations

Inlet Sets	Outlet Sets	Solution Predicts
velocity	static pressure	inflow static pressure
total pressure	velocity	outflow pressure inflow velocity
total pressure	static pressure	system mass flow

Table 1.1 Common boundary condition combinations

along with the global flow quantity which is estimated as part of the solution for each combination.

Domain Models

In incompressible flow fields the actual pressure level does not play any role in establishing the flow field - it is pressure differences which are important. The solver calculates these pressure differences with respect to a reference pressure. Solution fields are in relative pressure terms but absolute pressure (relative pressure plus reference pressure) is used for equation of state calculations.

In turbomachinery applications it is convenient to analyse the flow in a rotating reference frame. In this case, the domain is in a rotating reference frame and its axis of rotation and rotation rate must be specified.

Fluid Models

Heat Transfer Model: options include:

- **None:** no temperature field is computed (not an applicable option for ideal gases),
- **Isothermal:** a constant temperature field is used,
- **Thermal Energy:** a low speed (neglecting kinetic energy effects) form of the enthalpy conservation equation is computed to provide a temperature field, and
- **Total Energy:** a high speed form for conservation of energy including kinetic energy effects is computed.

Turbulence Model: options include:

- **None:** laminar flow simulation,
- **k-Epsilon:** the accepted state-of-the-art turbulence model involves the solution of two transport equations,
- **Shear Stress Transport:** a variant of the k-Epsilon model that provides a higher resolution solution in near wall regions, and
- **SSG Reynolds Stress:** a second moment closure model that explicitly solves transport equations for all six components of the turbulent stress tensor and that requires significantly more computing resources than the two equation variations.

Turbulent Wall Functions: are required to treat the transition to laminar like flow close to solid walls. The wall treatments are tied to the turbulence model choice. The scalable wall function method used with the k-Epsilon turbulence model is a variant of the standard wall function method. The

scalable wall function method automatically adjusts the near wall treatment with mesh spacing in the near wall region.

1.5.2 Initialization

The algebraic equation set that must be solved to find the velocity and pressure at each mesh point is composed of nonlinear equations. All strategies for solving nonlinear equation sets involve iteration which requires an initial guess for all solution variables. For a turbulent flow, sufficient information must be provided so that the following field values can be set (initialized) at each mesh point:

- velocity vector (3 components),
- fluid pressure,
- turbulent kinetic energy, and
- dissipation rate of turbulent kinetic energy.

CFX-Pre provides a default algorithm for calculating initial values based on interpolating boundary condition information into the interior of the domain. This default algorithm is adequate for many simulations.

The initial conditions can significantly impact the efficiency of the iterative solution algorithm. If there are condition difficulties then values or expressions (with **CEL**) can be used to provide intial conditions that:

- match the initial conditions to the dominant inlet boundary conditions, and
- align the flow roughly with the major flow paths from inlet regions to outlet regions.

1.5.3 Output Control

For many cases, especially those with a strong emphasis on fluid mechanics, it is necessary to output additional fields to the `res` output data file. For example, it is often worthwhile to output the turbulent stress fields throughout the flow domain, wall shear stresses on all boundary walls, and gradient operations applied to all primary solution variables.

1.5.4 Simulation Type

The numerical formulations for steady and transient flows differ slightly. The focus is on steady flow simulations, however transient evolution, with no transient accuracy, is used in the iterative solution algorithm. Each iteration is treated as a step forward in time.

1.5.5 Solver Control

The numerical methods operation used in the equation set solver are largely fixed, however some aspects of the numerical methods must be explicitly set by the user:

1. the choice of discretization scheme,
2. the time step size for the flow evolution, and
3. the criteria for stopping the iterative process.

The variation of velocity, pressure, etc. between the mesh points (element nodes) has to be approximated to form the discrete equations. These approximations are classified as the *discretization scheme*. The discretization scheme for approximating advective transport flows (listed in order of increasing accuracy) options are:

- Upwind - a constant profile between nodes,
- Specified Blend Factor- a blend of upwind and high resolution, and
- High Resolution - a linear profile between nodes.

In choosing a discretization scheme, accuracy is obviously an important consideration. Increasing the accuracy of the discretization often slows convergence: sometimes to the extent that the solution algorithm does not converge.

The choice of time step size for the flow evolution plays a big role in establishing the rate of convergence. Good results are usually obtained when the physical time step size is set to approximately 30% of the average residence time (or cycle time) of a fluid parcel in the flow domain. This residence time is referred to as the *global time scale*.

The initial guesses for the velocity, pressure, turbulent kinetic energy, and dissipation rate nodal values will not necessarily satisfy the discrete algebraic equations for each node. If the initial nodal values are substituted into the discrete equations there will be an imbalance in each equation which is known as the equation *residual*. As the nodal values change to approach the final solution, the residuals for each nodal equation should decrease.

The iterative algorithm will stop when either the maximum number of iterations is reached or when the convergence criterion is reached (whichever occurs first). The convergence criterion is a convergence goal for either the maximum normalized residuals or the root mean square (RMS) of the normalized residuals. Note that the residuals are normalized to have values near one at the start of the iterative process.

1.6 CFX Solver Manager: Solver Operation

A solver run requires a definition file to define and initialize the run. Table (1.2) shows how `def` and `res` files can be used to define different runs.

Definition File	Initial File	Use
`def`		Start from simple initial fields
`res`		Continue solution for further convergence
`def`	`res`	Restart from existing solution with new flow model

Table 1.2 Input file combinations

1.6.1 Monitoring the Solver Run

The solution of the algebraic equation set is the component of the code operation which takes the most computer time. Fortunately, because it operates in a batch

mode, it does not take much of the user's time. The operation of the solver should, however, be monitored and facilities are provided for this.

Table (1.3) shows typical solver diagnostic output listing the residual reduction properties for the first few time steps (iterations) of a solver run.

```
=====================================================================
OUTER LOOP ITERATION =    1                    CPU SECONDS = 1.79E+00
---------------------------------------------------------------------
|       Equation       | Rate | RMS Res | Max Res |  Linear Solution |
+----------------------+------+---------+---------+------------------+
| U-Mom                | 0.00 | 2.2E-02 | 1.5E-01 |       2.4E-02  OK|
| V-Mom                | 0.00 | 3.6E-08 | 1.2E-07 |       9.7E+03  ok|
| W-Mom                | 0.00 | 9.5E-20 | 4.7E-19 |       1.4E+12  ok|
| P-Mass               | 0.00 | 2.8E-03 | 4.6E-02 | 10.2  4.9E-02  OK|
+----------------------+------+---------+---------+------------------+
| K-TurbKE             | 0.00 | 1.8E-01 | 6.0E-01 |  5.7  6.0E-05  OK|
+----------------------+------+---------+---------+------------------+
| E-Diss.K             | 0.00 | 1.6E-01 | 1.0E+00 |  7.1  1.5E-06  OK|
+----------------------+------+---------+---------+------------------+

=====================================================================
OUTER LOOP ITERATION =    2                    CPU SECONDS = 2.81E+00
---------------------------------------------------------------------
|       Equation       | Rate | RMS Res | Max Res |  Linear Solution |
+----------------------+------+---------+---------+------------------+
| U-Mom                | 6.37 | 1.4E-01 | 5.0E-01 |       8.5E-04  OK|
| V-Mom                |99.99 | 9.1E-03 | 1.0E-01 |       2.8E-02  OK|
| W-Mom                |99.99 | 2.8E-11 | 3.2E-10 |       6.6E+03  ok|
| P-Mass               | 0.82 | 2.3E-03 | 3.4E-02 | 10.2  6.8E-02  OK|
+----------------------+------+---------+---------+------------------+
| K-TurbKE             | 0.21 | 3.7E-02 | 1.7E-01 |  5.8  2.1E-04  OK|
+----------------------+------+---------+---------+------------------+
| E-Diss.K             | 0.24 | 3.8E-02 | 3.4E-01 |  7.1  1.1E-05  OK|
+----------------------+------+---------+---------+------------------+
```

Table 1.3 Typical convergence diagnostics

For each field variable equation set, the following information is output each time step:

Rate the convergence rate

$$\texttt{Rate} = \frac{\texttt{RMS res. current time step}}{\texttt{RMS res. previous time step}}$$

which should typically be 0.95 or less,

RMS Res the root mean square of the nodal normalized residuals,

Max Res the maximum nodal normalized residual in the flow domain, and

Linear Solution after each equation set is linearized, an estimate of the solution of the resulting linear equation set is obtained and statistics on this solution are reported:

Work Units a measure of the effort required to obtain the solution estimate,

Residual Reduction the amount that the linear solver has reduced the RMS residual of the linear equation set, and

Status an indicator of the linear solver performance:

OK residual reduction criteria met,

ok residual reduction criteria not met but converging,

F solution diverging,

***** residual increased dramatically and solver terminates due to floating point number overflow error.

Some of the above information is displayed graphically in the monitor window so that the solver execution can be monitored.

When execution is complete the final results are written to the `res` file. In addition, all of the information pertinent to the operation of the solver is output to the `out` file, including:

- the CPU memory or storage requirements,
- the physical flow model,
- grid summary,
- estimate of the global length, speed, and time scales based on the initial fields,
- the convergence diagnostics,
- estimate of the global length, speed, and time scales based on the final fields,
- the fluxes of all conserved quantities through the boundary surfaces (these should balance to 0.01% of the maximum fluxes), and
- the computational time required to obtain the solution.

1.7 CFX-Post: Visualization and Analysis of Results

To the typical user of CFD, the generation of the velocity and pressure fields is not the most exciting part. It is the ability to view the flow field that makes CFD such a powerful design tool. CFX-Post has capabilities for visualizing the results in graphics objects, for using calculation tools, and for controlling the post-processing state.

1.7.1 Objects

Three types of objects can be created: geometrical, flow visualization, and viewer augmentations. Geometrical objects include:

Point: a point in the domain. Often used to probe flow properties within the domain;

Line: a straight line between two points on the line. Intermediate points on the line can be set at intesections with mesh faces, referred to as a *cut* line, or uniformly spread along the line, referred to as a *sample* line;

Plane: a flat plane in the domain. Like a line object, a plane object can be either a cut plane or a sample plane;

Isosurface: the surface along which some scalar field property has a constant value;

Polyline: a piecewise continuous straight line between a series of points. The line can be derived from the intersection of a boundary and another geometrical object or as a contour line; and

User Surface: a surface derived in a manner similar to a polyline object.

The definition of each of these objects involves choosing from a range of options for each atttribute including: location, line colour, etc. The object edit panels outline the possible options for each attribute of the object. See the on-line help for further information on each of these objects and their generation.

Flow visualization objects allows exploration of the velocity, pressure, temperature fields. For each flow property calculated at a node there are two fields: *hybrid* and *conservative*. At interior nodes the two fields are identical. For nodes on the wall there are two values for every flow property: the value implied by the boundary condition and the average value in the sub-element region around the node. The first value is the hybrid value and the second value is the conservative value. For a node on a solid wall, the hybrid velocity will be zero and the conservative velocity will typically be non-zero.

Common flow visualization objects include:

Wireframe: a singleton object automatically created to show the surface mesh on the flow domain. The *edge angle* controls how much of the surface mesh is drawn. An element edge is drawn if the angle between the two adjacent element faces is bigger than the edge angle. For rectangular geometries and meshes an edge angle of 30 ° ensures that the outside edges of the domain are drawn and an edge angle of 0 ° ensures that all mesh edges are drawn;

Contour: lines of a constant scalar value on a specified surface (like elevation lines on a topological map). The appearance of the plot is controlled by the rendering options. For example, with the smooth shading option turned on a fringe plot (areas between contour lines are filled with colour) is drawn and with no shading only the contour lines are drawn;

Vector: the field of vectors in a region;

Streamline: lines that are parallel to the local velocity vectors. Typically streamlines are started on an inlet surface and will extend to an outlet surface; and

Particle Tracks: If particle paths (an advanced feature) were calculated as part of the simulation then these paths can be drawn.

Again, see the on-line help for further information on each of these objects and their generation.

Common **Viewer** augmentation objects include:

Legend: a scale legend to associate property values with colours;

Text: a piece of text;

Clip Plane: a flat plane that is used to reduce the portion of the domain shown in the viewer; and

Instancing Transformation: for repeating views to show full geometries when periodic or symmetry conditions have been used to model a portion of a domain.

1.7.2 Tools

Common post-processing tools include:

File/Export ...: allows the export of portions of the results in a space-separated tabular form. This is useful for exporting data to programs like Excel for graphing, etc.;

File/Print ...: allows the export and printing of the graphical image in the viewer window. Images saved in encapsulated postscript (`eps`) or portable network graphics (`png`) formats can be easily inserted into reports and presentations;

Tools/Calculator: is a powerful tool for carrying out a wide range of mathematical calculations including a range of integration and averaging calculations;

Create/Expression: allows the input of **CEL** expressions. To learn the expression syntax right, mouse click in the **Definition** box within the **Expression Editor** panel to get a list of possible variables, expressions, locators, functions and constants that can be input into a new expression. There are two type of functions: **Functions/CFX- Post** functions for performing operations on scalar fields and **Functions/CEL** functions for conventional mathematical operations; and

Create/Variable: allows the calculation of new field variables. New variables can be defined in terms of defined expressions or by entering new expressions directly into the variable definition. For example, calculating local pressure coefficients based on the maximum speed in the domain, backstep (i.e. $C_p \equiv \frac{(p-p_{ref})}{\frac{1}{2}\rho V_{max}^2}$) can be done with the expression:

```
(Pressure - 0 [Pa])/(0.5*Density*maxVal(sqrt(Velocity u^2 + Velocity v^2))@backstep^2)
```

for a variable called Cp

1.7.3 Controls

A set of controls can be used to save a particular post-processing setup or *state* so that it can be repeated. This is useful when comparing different simulations to see the impact of a design change on flow properties. Common controls include:

Camera: The image size, orientation, perspective, etc. in the **Viewer** window is associated with a Camera viewer (imagine that the user moves a camera around in space to create a two-dimensional image of a three-dimensional object in the camera's viewer). Camera views can be saved, deleted, and refreshed with the buttons on the right top of the **Viewer** window. Note that the camera does not determine which graphics objects are visible;

Session: Provides a mechanism for recording and saving a series of operations. This is useful for users interested in learning power programming of CFX-Post (see on-line help for more information);

State: Saves all information (views, graphical objects, expressions and variable) in a `cst` state file. Loading a state file will automatically re-generate all views, objects, expressions, and variables from a previous CFX-Post session.

1.8 Commands for Duct Bend Example

As indicated in Section 1, the following conventions will be used to indicate the various commands that should be invoked:

Menu/Sub-Menu/Sub-Sub-Menu Item chosen from the menu hierarchy at the top of a main panel or window,

Button/Tab Command Option activated by clicking on a button or tab,

Link description Click on the description to move by a link to the next step/page,

Name *value* Enter the *value* in the named box,

Name ▼ **selection** Choose the selection(s) from the named list,

Name Panel or window name,

☐ **Name** On/off switch box, and

◯ **Name** On/off switch circle (radio button).

1.8.1 Geometry Model

The first phase uses DesignModeler. The commands listed below will create a solid body geometry that will represent the flow domain:

- set up the project directory and page;
- set up a plane for sketching on;
- sketch in the flow path through the duct, Figure 1.4;
- generate the solid body geometry;

To begin:

- Create a new folder `N:/Ductbend` to be the working directory for the project files.
- Open ANSYS Workbench from **Start/Programs/Engineering/ANSYS 9.0/ANSYS Workbench**. To open a new project:
  1. In the **Start** window, select Empty Project in the **New** panel,
  2. From the menu bar choose **File/Save As ...** to create a project file in your working directory, and
  3. In the **Save As** window fill in **File name:** Ductbend.wbdb and click the **Save** button.
- From the menu bar choose **Tools/Options ...** to change the length units options. In the **Options** window expand the + DesignModeler entity and select ☐ **Grid Defaults (Meters)** in the tree view on the left. Set **Minimum Axes Length** 0.5 and **Major Grid Spacing** 0.1. Click **OK** to save the options and close the window,
- Choose the Create DesignModeler geometry task,

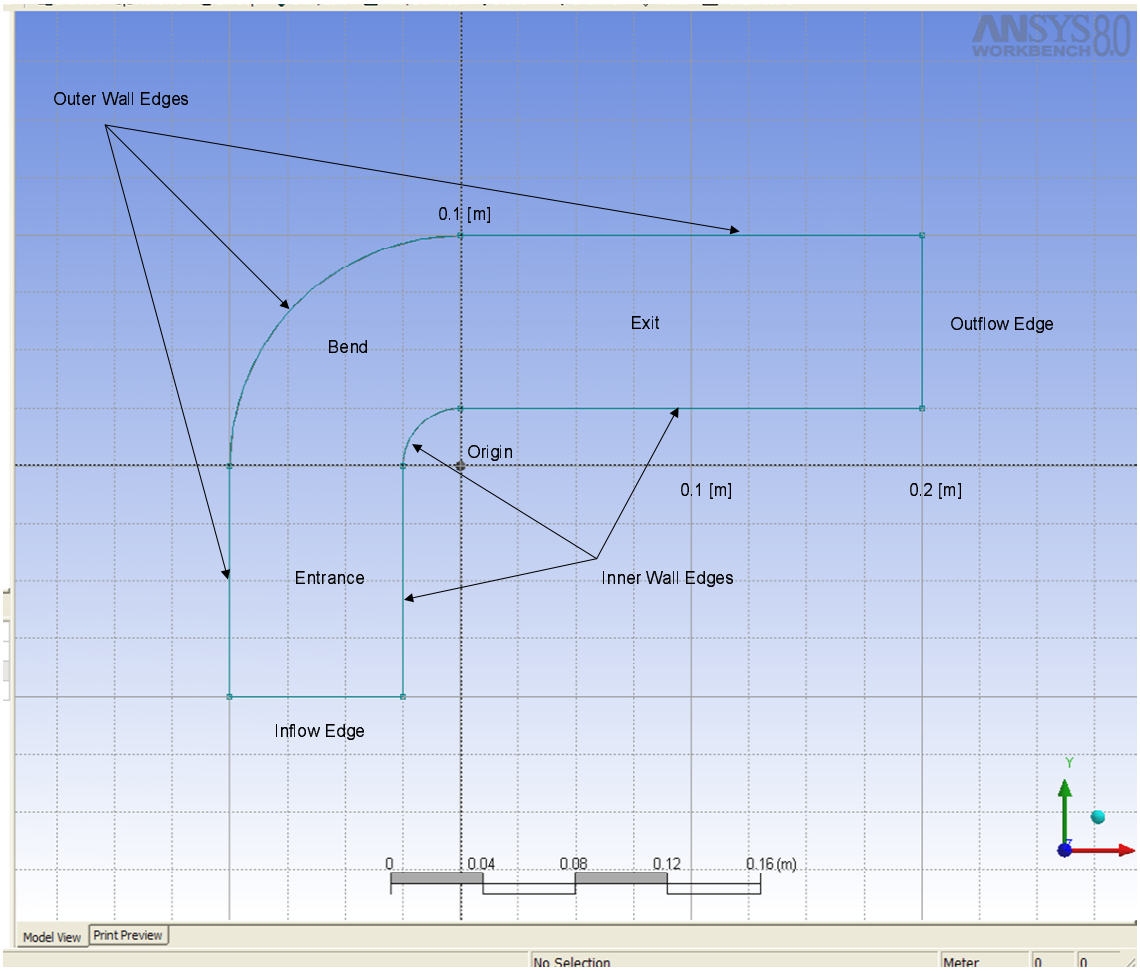

Fig. 1.4 Wireframe of the final sketch of the duct bend flow path.

- Check that the desired length unit is ◯ **Meter** is selected and click Ok in the units window that appears,
- In the Tree View, select the XYPlane entity and then click the **New Sketch** icon to create the Sketch1 entity as a component of the XYPlane.
- To start the sketching , select the **Sketching** tab
- To draw in the 2D sketch of the duct flow path (Figure 1.4)
  1. Select the **Sketching** tab; and
  2. Click on the Z coordinate of the triad in the lower right corner of the Model View.
- Select the **Draw** toolbox and:

1. With the **Arc by Center** tool sketch the inner wall bend shape.
   (a) Place the cursor over the origin (watch for the P constraint symbol) and left mouse button click.
   (b) Move the cursor to the left along the X axis. With the C constraint visible click the left mouse button to put the start point of the arc on the X axis.
   (c) Sweep the cursor clockwise until the C constraint appears at the Y axis. Click the left mouse button.[2]
2. Switch to the **Dimensions** toolbox to size the inner wall bend radius:
   (a) Select the **Radius** tool;
   (b) Select a point on the arc. Then move the cursor to the inside of the arc near the origin. Click to complete a dimension which is labelled R1.
   (c) In the Details View notice that R1 is shown under the **Dimensions** title.
   (d) Change the value of R1 to 0.025 [m]. Notice the arc radius changes automatically. If the dimension is poorly placed on your sketch you can use the **Move** tool to correct the placement.
3. Switch back to the **Draw** toolbox to sketch the inner entrance wall:
   (a) With the **Line** tool selected, place the cursor in the lower left quadrant of the XY plane near the arc. Click the left mouse button. Move the mouse cursor down to create a vertical line. Look for the V constraint symbol and click the left mouse button;
   (b) Switch to the **Dimensions** toolbox to size the inner entrance wall length:
      i. Select the **General** tool;
      ii. Select a point near the centre of the line. Then move the cursor to the right. Click to complete a dimension which is labelled V2.
      iii. In the Details View change the value of V2 to 0.10 [m].
   (c) To join the inner entrance wall and the inner wall bend switch to the **Constraints** toolbox;
      i. Select the **Coincident** tool;
      ii. Select the upper end of the entrance inner wall with a left mouse button click. The square end marker should be yellow;
      iii. Select the square end marker of the arc that lies on the X axis with a left mouse button click. The inner entrance wall should join the inner wall bend.
4. To draw a line across the inflow (entrance):
   (a) Use the **Line** tool in the **Draw** toolbox;
   (b) Place the cursor over the bottom end point of the entrance inner wall and notice that a P constraint symbol appears. Left mouse

[2]Notice that the drawing instruction steps are provided in the lower left corner.

button click to select this point and then move the cursor to the left and click while the H constraint symbol is visible.

(c) Use the **General** tool in the **Dimensions** toolbox:

(d) Select a point near the centre of the line. Then move the cursor down. Click to complete a dimension which is labelled H3.

(e) In the Details View change the value of H3 to 0.075 [m].

5. Repeat the procedure used for the entrance inner wall to draw the exit inner wall:
   (a) Draw a horizontal line in the upper right XY quadrant near the end point of the inner wall bend;
   (b) Set the length of the line to 0.20 [m] with the **General** tool from the **Dimensions** toolbox;
   (c) Join the exit inner wall to the inner wall bend with the **Coincident** tool from the **Constraints** toolbox.
6. Draw the outer entrance wall with the **Line** tool. Start at the outer (left) end point of the inflow edge (look for the P constraint symbol) and draw a vertical line that is coincident (C) with the X axis;
7. Draw the outer bend wall with the **Arc by Center** tool. Put the center at the origin, make the start point at approximately 20° above the X axis in the upper left quadrant, and make the end point coincident (C) with the Y axis. Use the **Coincident** constraint tool to join the start point of the arc to the end point of the outer entrance wall;
8. Draw the outer exit wall with the **Line** tool. Draw a horizontal (H) line coincident (C) with the Y axis above its final desired location. Use the **Coincident** constraint tool to join this line to the end of the outer bend wall. Use the **Equal Length** constraint tool to make the outer exit wall the same length as the inner exit wall; and
9. Draw a line from the end of the outer exit wall to the end of the inner exit wall to form the outflow edge. Make sure that the end points are coincident (P).

The sketch should now be an enclosed contour on the XYPlane.

- To create the three dimensional solid body:
  1. Switch to the Tree View by selecting the **Modeling** tab;
  2. Click on the **Extrude** button to create the Extrude1 feature . In the Details View:
     * Check **Base Object** Sketch1,
     * Select **Operation** ▼ Add Material,
     * Select **Direction** ▼ (NONE) Normal,
     * Set **FD1, Depth (**> 0**)** 0.02,
     * Select **As Thin/Surface?** ▼ No, and

   * Select **Merge Topology?** ▼ Yes.
3. Click on the **Generate** button to create a Solid. Use the isometric view in the Model View to check that you have a three dimensional solid grey body.

- Choose **File/Save As ...** and set **File name:** Ductbend.agdb in the **Save As** window. Click **Save** to close window.
- Return to the Project page by clicking on the **Ductbend [Project]** tab at the top left corner of the window.

1.8.2 Mesh Generation

The second phase uses CFX-Mesh. The commands listed below will generate a discrete mesh in the flow domain:

- name the surfaces (faces) of the solid geometry to ease boundary condition specification;
- specify the properties of the mesh in the interior of the flow domain and close to solid walls;
- preview the surface mesh to check for anomalies; and
- create the geometry file with volume mesh information.

- Choose the Generate CFX Mesh DesignModeler Task; Notice the three primary areas: Model View, Tree View, and Details View.
- The surfaces (faces) of the solid are labeled as regions for ease of attaching the boundary conditions . To attach a region entity at the inflow (see Figure 1.4):
  1. Right mouse click over Regions in the Tree View and select **Insert** ▼ Composite 2D Region to create a new region entity;
  2. Left click over the new entity's name, Composite 2D Region 1, and edit the region name to inflow surface;
  3. In the Model View put the mouse cursor over the inlet surface area and left mouse click. The inlet surface area should turn green; and
  4. In the Details View click on **Location** ▼ Apply;
- Repeat to create a region entity called outflow surface. Notice that if you left mouse button click on inflow surface or the outflow surface in the Tree View that the resulting region in the Model View turns green.
- Create a region entity called inner wall. This region is composed of three primitive surfaces. To select a set of surfaces or faces hold the "Ctrl" key down while clicking on the component surfaces in the Model View;
- Repeat to create an entity called outer wall;
- For a region called front surface use the Z view of the Model View when selecting the surface. Notice that there are two parallel planes in the lower left corner of the model view. The front most of these planes should be outlined in red.

- Do not set up a region for the back surface. It will remain as the single surface in the Default 2D Region.
- In the Tree View, select Options to see the mesh options in the the Details View. Set **Surface Meshing** ▼ Advancing Front and **Meshing Strategy** ▼ Extruded 2D Mesh;
- In the Tree View, expand the + Spacing entity;
- Select the Default Body Spacing entity to open the Body Spacing Details View. Set **Maximum Spacing [m]** 0.0075.
- Right mouse click on the Extruded Periodic Pair entity;
  - In the Model View select the front surface and then click **Location 1** Apply in the Details View;
  - In the Model View select the back surface (remember to use the location planes in the lower left corner of the Model View) and then click **Location 2** Apply in the Details View; and
  - Set **Periodic Type** ▼ Translation;
- In the Tree View, right mouse click on Inflation and select **Insert/Inflated Boundary** to create an Inflated Boundary entity. Select the three surfaces of the inner wall in the Model View for the Location and set **Maximum Thickness [m]** 0.0075;
- Repeat to create an Inflated Boundary of **Maximum Thickness [m]** 0.0075 on the outer wall;
- In the Tree View expand the + Preview entity. Right mouse click on Default Preview Group and select **Generate This Surface Mesh**. An Overall Progress window should open in the lower left corner and after a short time you should see a mesh of triangles on the surfaces of the solid. There will be some gaps on the front and back surfaces near the inner and outer walls where the inflated boundary meshing is used;
- Click the **Generate a volume mesh for the current problem** icon on the top row of icons/buttons. In the Windows file window set **File name:** Ductbend.gtm. Again a progress window will open in the lower left corner. When this window closes go to the Tree View and select Errors to ensure that no errors are reported in the Details View.
- To close this phase, select **File/Save As ...** and set **File name:** Ductbend.cmdb.
- Return to the Project page by clicking on the **Ductbend [Project]** tab at the top left corner of the window.

1.8.3 Pre-processing

The first CFD phase is preprocessing. In this phase the complete CFD model (mesh, fluids, flow processes, boundary conditions, etc.) is defined and saved in a hierarchical database. After opening `CFX-Pre`, the commands listed below will accomplish the following steps:

- link the mesh `gtm` file to a CFX flow project,
- set simulation type to steady state,
- establish a fluid with nominal properties of water,
- specify the region through which the fluid will flow, the fluid (nominally water), and the physical models (fluid flow, no heat transfer, turbulence, standard k-e model with scalable wall functions),
- set up and attach the rough wall boundary condition, the inlet boundary condition, the outlet boundary condition, and the symmetry boundary conditions,
- set the global initial conditions,
- set the solver controls for discretization scheme, time step type, and convergence criteria,
- set the output variable list, and
- write the complete CFD model definition to a `def` file.

To accomplish these steps execute the following commands:

- Highlight the mesh file, `Ductbend.gtm`, in the Project page. Choose Create CFD Simulation with Mesh Advanced CFD Task;
- After a short wait the **CFX-Pre** page will open. This page is similar to the **CFX-Post** page. There are three main areas: the menus and command buttons at the top, the Viewer window at the right, and the database panels at the left;
- Select the **Regions** tab to see the region database:
  - Check that ☐ **Assembly** entity under the Mesh Assemblies tree is on. Click on the name of the entity, `Assembly`, to see the mesh on the duct wireframe model;
  - To see the mesh on an individual region click the named regions in the Composite Regions tree. Click through each region in the tree;
  - Expand + the inner wall region to see the three primitive faces that make up the region. Each primitive face has a name of the form Fx.By - face x of body y where only the leading letter (F or B) is shown.
  - The (back surface was imported into CFX-Pre attached to the Default 2D Region. If you would like to correct this naming anomaly click **Create New Object** icon on the right of the **Region** panel. A small **Create Object** panel will open. Fill in **Name** back surface (assuming that this is the missing surface) and click **OK** to open the **Region Editor** panel will open up where you set:
    - **Combination** Alias
    - **Dimension** ▼ All
    - Select Default 2D Region to put into the **Region List** panel list; and
    - Click **Ok** to close panel.

- Click the **Define the Simulation Type** button to open the **Simulation Type** panel. Check that **Option** ▼ Steady State is set and then click **Ok**.
- Click **Materials** tab in the database panel to open up a list of available materials. To define a new material click the **Create New Object** icon. In the **Create Material** panel, fill in **Name** Water nominal and click **OK** to open a panel with two tabs:
  - Click the **Basic Settings** tab and set:
  - **Option** ▼ Pure Substance,
  - **Material Group** ▼ Constant Property Liquids,
  - ☐ **Thermodynamic State** on and **Thermodynamic State** ▼ Liquid.
  - Click the **Material Properties** tab and set:
  - **Option** ▼ General Material,
  - expand **Equation of State** +,
  - **Option** ▼ Value,
  - **Density** 1000 ▼ kgm^-3,
  - expand **Transport Properties** +,
  - ☐ **Dynamic Viscosity** on and **Dynamic Viscosity** 0.001 ▼ kgm^-1s^-1,

  and then click **Ok**.
- Click the **Domains** button. A small **Create Domain** definition panel will open. Fill in **Name** Ductbend and click **OK** to define the domain object. An **Edit Domain: Ductbend** panel will open with several tabbed sub-panels.
  - On the **General Options** sub-panel, set:
    - **Location** ▼ Assembly,
    - **Domain Type** ▼ Fluid Domain,
    - **Fluids List** ▼ Water nominal,
    - ☐ **Particle Tracking** off,
    - **Reference Pressure** 1 ▼ atm,
    - **Buoyancy Option** ▼ Non Buoyant, and
    - **Domain Motion Option** ▼ Stationary.
  - on the **Fluid Models** sub-panel set:
    - **Heat Transfer Model Option** ▼ None,
    - **Turbulence Model Option** ▼ k-Epsilon,
    - **Turbulent Wall Functions Option** ▼ Scalable,
    - **Reaction or Combustion Model Option** ▼ None, and
    - **Thermal Radiation Model Option** ▼ None,

  - on the **Initialization** sub-panel ensure that ☐ **Domain Initialization** is off and then click **Ok** to close the panel. Notice that the Domain: Ductbend is now listed in the **Physics** database tree. If you double-click on this object in the list then the **Edit** panel will reappear.

- Click the **Create a Boundary Condition** button to open a definition panel. Fill in **Name** INFLOW SURFACE and click **OK**. In the **Edit Boundary: Inflow surface in Domain: Ductbend** panel:
  - under the **Basic Settings** tab set:
    - **Boundary Type** ▼ Inlet, and
    - **Location** ▼ inflow surface,
  - and under the **Boundary Details** tab set:
    - **Flow Regime Option** ▼ Subsonic,
    - **Mass and Momentum Option** ▼ Normal Speed,
    - **Normal Speed** ▼ 3 ▼ ms^-1,
    - **Turbulence Option** ▼ Intensity and Length Scale,
    - **Fractional Intensity** 0.05,
    - **Eddy Length Scale** 0.0075 ▼ m,

  and then click **Ok** to close the panel and create the new boundary object. Notice that incoming arrows appear on the inlet surface in the **Viewer** window and the boundary object is listed in the **Physics** database tree. Clicking on **Boundary: INFLOW SURFACE** object in the **Physics** database causes the inflow surface mesh to be outlined with green in the **Viewer** window.
- Click the **Create a Boundary Condition** button to open the definition panel. Fill in **Name** OUTFLOW SURFACE and click **OK**. In the **Edit Boundary: Outflow surface in Domain: Ductbend** panel:
  - under the **Basic Settings** tab set:
    - **Boundary Type** ▼ Outlet, and
    - **Location** ▼ outflow surface,
  - and under the **Boundary Details** tab set:
    - **Flow Regime Option** ▼ Subsonic,
    - **Mass and Momentum Option** ▼ Average Static Pressure,
    - **Relative Pressure** 0 ▼ Pa,

  and then click **Ok** to close the panel and create the new boundary object.
- Click the **Create a Boundary Condition** button to open the definition

panel. Fill in **Name** FRONT SURFACE and click **OK**. In the **Edit Boundary: Front Surface in Domain: Ductbend** panel:
  * under the **Basic Settings** tab set:
    - **Boundary Type** ▼ Symmetry, and
    - **Location** ▼ front surface,

  and then click **Ok** to close the panel and create the new boundary object.
- Click the **Create a Boundary Condition** button to open the definition panel. Fill in **Name** BACK SURFACE and click **OK**. In the **Edit Boundary: Back surface in Domain: Ductbend** panel:
  * under the **Basic Settings** tab set:
    - **Boundary Type** ▼ Symmetry, and
    - **Location** ▼ back surface,

  and then click **Ok** to close the panel and create the new boundary object.
- Click the **Create a Boundary Condition** button to open the definition panel. Fill in **Name** INNER WALL and click **OK**. In the **Edit Boundary: Inner wall in Domain: Ductbend** panel:
  * under the **Basic Settings** tab set:
    - **Boundary Type** ▼ Wall, and
    - **Location** ▼ inner wall,
  * and under the **Boundary Details** tab set:
    - **Wall Influence on Flow Option** ▼ No Slip,
    - ☐ **Wall Velocity** off,
    - **Wall Roughness Option** ▼ Rough Wall,
    - **Roughness Height** 0.0001 ▼ m,

  and then click **Ok** to close the panel and create the new boundary object.
- Click the **Create a Boundary Condition** button to open the definition panel. Fill in **Name** OUTER WALL and click **OK**. In the **Edit Boundary: Outer wall in Domain: Ductbend** panel:
  * under the **Basic Settings** tab set:
    - **Boundary Type** ▼ Wall, and
    - **Location** ▼ outer wall,
  * and under the **Boundary Details** tab set:
    - **Wall Influence on Flow Option** ▼ No Slip,
    - ☐ **Wall Velocity** off,
    - **Wall Roughness Option** ▼ Rough Wall,
    - **Roughness Height** 0.0001 ▼ m,

and then click **Ok** to close the panel and create the new boundary object.

- Click on the **Define the Global Initial Conditions** button to open the initialization panel. Make sure that ☐ **Turbulence Eddy Dissipation** is on. You can accept ▼ Automatic initialization for all properties and click **Ok**.
- Click on the **Define the Solver Control Criteria** button to open the panel. Under the **Basic Settings** tab set:
  - **Advection Scheme Option** ▼ Upwind,
  - **Timescale Control** ▼ Physical Timescale,
  - **Physical Timescale** 0.04 ▼ *s* (30% of the average residence time of a fluid parcel inside the flow domain),
  - **Max No. Iterations** 50,
  - **Residual Type** ▼ MAX,
  - **Residual Target** 1.0e-3,

  and then click **Ok**.
- Click the **Create Output Files and Monitor Points** button to open the panel. In the **Results** tab panel set:
  - **Option** ▼ Full,
  - ☐ **Output Variable Operators** on and choose ▼ All, and
  - ☐ **Output Boundary Flows** on and choose ▼ All,

  and then click **Ok**.
- Click the **Write a Solver File** to open the panel. Accept the default filename, `Ductbend.def` and **Operation** ▼ Start Solver Manager. Click **OK**.

1.8.4 Solver Manager

The CFX-Solver window will open after CFX-Pre closes. In the `Define Run` panel set:

- **Definition File** ductbend.def (NOTE: If restarting a partially converged run, you would enter the name of the most current results file),
- **Type of Run** ▼ Full, and
- **Run Mode** ▼ Serial,

and then click **Start Run**.

After a few minutes execution should begin. Diagnostics will scroll on the terminal output panel and the equation RMS residuals will be plotted as a function of time step. After the first few time steps, the residuals should fall monotonically. Execution should stop after 29 time steps. In the **CFX-5 Solver Finished Normally** window click **OK**.

1.8.5 Post-processing

The most interesting step is the analysis of the results. To illustrate this step the commands listed in the next paragraph step through the following tasks:

- load the results from the `res` file for post-processing,
- create a vector plot on one of the symmetry boundary planes,
- save an image of the plot,
- create a vorticity variable,
- create a fringe plot of the vorticity field,
- create a line and export the velocity data along the line,
- export the inner wall pressure and wall shear stress distribution,
- probe the velocity field, and
- save the visualization state.

To accomplish these tasks click the **CFX-Post** tab in bottom left corner and then:

- Choose **File/Load Results**, and select the `Ductbend_001.res` results file.
- Choose **Create/Object/Vector**, accept **Name** Vector 1, and click **OK** to define a vector object and open an edit panel. In the panel set:
  - **Locations** ▼ FRONT SURFACE,
  - **Variable** ▼ Velocity,
  - ◯ **Hybrid** on,
  - **Projection** ▼ None,
  - **Reduction** 1 (plots vector at every mesh point),
  - ☐ **Visibility** on

  and click **Apply**. The vector plot should appear in the **Viewer** window and the vector object is listed in the **Object** database panel. Under the **View Control** branch in the **Object** tree, turn ☐ **DEFAULT LEGEND** off and back on to remove and then replace the scale legend. Under the **Regions** branch turn ☐ **Wireframe** on. Orthographic projection will work best for two-dimensional views. Notice that if you double-click on an object in the **Object** tree then an edit panel opens up for that object.
- Choose **File/Print ...** and in the **Print** panel set:
  - **File** vectorplot.png, and
  - **Format** ▼ PNG

  followed by **Print**. The plot will printed to the file `vectorplot.png` in your working directory. You can import this file into other documents.
- Choose **Create/Variable** and in the **New Variable** definition window set **Name** Vorticity and click **OK**. In **Vorticity** edit panel (lower left):
  - set **Method** ▼ Expression,
  - set ◯ **Scalar** on,

  - fill in **Expression** Velocity v.Gradient X - Velocity u.Gradient Y, and
  - click **Apply**.

- Choose **Create/Object/Contour**, accept **Name** Contour 1, and click **OK** to define a fringe/contour plot object and open an edit panel. In the panel set:
  - **Locations** ▼ FRONT SURFACE,
  - **Variable** ▼ Vorticity,
  - ☐ **Visibility** on

  and click **Apply**. The fringe plot should appear in the **Viewer** window.
- Choose **Create/Object/Line**, accept **Name** Line 1, and click **OK** to define a line object and open an edit panel. Use **Method** ▼ Two Points, set Point 1 to (0.0,0.1,0.01), set Point 2 to (0.0,0.025,0.01) (notice that we put the line in the middle of the domain to avoid boundary effects), set the number of samples to 25, and click **Apply** to see the line (make sure that the visibility of the contour plot, etc. is turned off).
- Choose **File/Export ...** to open the `Export` panel where you can:
  - set **File** uvelocity.dat,
  - select Line 1 from the **Locator** list,
  - set ☐ **Export Geometry Information** on,
  - select (Ctrl key plus click) Velocity u and Velocity v from the **Select Variable(s)** list, and
  - click **Save** to write the data to a file in a space-separated format that can be imported into a conventional spreadsheet program for plotting or further analysis. Notice that this file includes x,y, and z values.
- Choose **Create/Object/Plane**, accept **Name** Plane 1, and click **OK** to define a plane object and open an edit panel. Use **Method** ▼ XY Plane with $Z = 0[m]$ and click **Apply** (set ☐ **Visibility** off to avoid clutter in the view).
- Choose **Create/Object/Polyline**, accept **Name** Polyline 1, and click **OK** to define a polyline object. In the edit panel use **Method** ▼ Boundary Intersection with **Boundary List** ▼ INNER WALL and **Intersects With** ▼ Plane 1 and then click **Apply**. This creates a line the follows the inner wall. You can follow the steps for export along a line to export the values of the pressure, total pressure, and wall shear stress along this line into the data file `wall.dat`.
- Choose **Create/Object/Point**, accept **Name** Point 1, and click **OK** to define a point object. Use **Method** ▼ XYZ and initialize the point to (0.10,0.04,0) before clicking **Apply**. Choose **Tools/Calculator** to open the

Function Calculator panel. Use **Function** ▼ probe, **Location** ▼ Point 1, **Variable** ▼ Velocity u.Gradient X (Note: Use the ... to get a list of all possible variables.) before clicking on the **Calculate** button. The result with units appears in the results box. Move the point around to probe other regions in the flow.

- Choose **File/Save State** and enter `tutorial1.cst` for the file name to save all of the information associated with the visualization and post-processing objects you have created in this session. You can load this state file (**File/Load State**) to recreate these objects and images in later sessions. This facility allows easy comparison of results between simulations.
- Return to the Project page and choose **File/Exit**. Select Yes to save highlighted files.

1.8.6 Clean Up

The last step is to remove unnecessary files created by CFX-5. This step is necessary to ensure that you do not exceed your disk quota. At the end of each session delete all files except:

- `*.agdb`, `*.cmdat`, `*.cmdb`, `*.wbdb`, `*.def` and `*_*.res` files.

If you no longer need your results but would like to be able to replicate them then you should delete all files except:

- `*.def` files.

After removing all unnecessary files, use the WinZip utility to compress the contents of your directory.